RO7

TECHNICAL PROGRESS REPORT NO.8

DEVELOPMENT OF THE INSTRUMENTATION AND MODELING FOR HEAT TRANSFER CHARACTERISTICS IN CFBC

TO

U.S. DEPARTMENT OF ENERGY
FEDERAL ENERGY TECHNOLOGY CENTER
P.O. BOX 10940, MS 921-118
PITTSBURGH, PA 15236-0940

FOR

PROJECT NO.: DE-FG22-95MT95013

BY

DR. SEONG W. LEE, PRINCIPAL INVESTIGATOR

MORGAN STATE UNIVERSITY
SCHOOL OF ENGINEERING
BALTIMORE, MD 21239
(phone) 443-885-3106

October 1999

ABSTRACT

This technical report summarizes the research conducted and results obtained during the period beginning April 1, 1999 to September 30, 1999.

Numerical simulation was used to predict the air velocity and pressure, temperature, and heat transfer characteristics in the CFB system.

The radial air velocities changed frequently because of the voidage and gas pressure drop that caused in the heat transfer probe inlet level (k=23). The lower pressure zone was formed near the chamber's center region. The higher pressure zone was formed at the surrounding of the secondary air flow.

The lower temperature profiles were shown to be near the CFB riser wall side. In the radial direction, the gas temperature nearest to the center region was higher than that near the wall zone. The heat flux was changed along the radial direction of the CFB riser.

The heat transfer probe was designed and installed in the CFB riser. The heat transfer rate at different axial/radial locations of the CFB riser will be measured by the heat transfer probe. A model will be proposed to predict the heat transfer coefficients along the axial and radial variation of the bed in the CFB system.

TABLE OF CONTENT

SECTION 1

NUMERICAL SIMULATION FOR THE CFB MODEL WITH HEAT TRANSFER

The numerical simulation was conducted to predict the air velocity and pressure, temperature, and heat transfer characteristics in the CFB system. The flow system and the solution domain were introduced in the previous report [1].

1.1 The Effect of Air Velocity

Figure 1 shows the 2-D velocity profiles at k=23 of the vertical direction in the slide plate which is the heat transfer probe inlet level. When the air flow reached the heat transfer probe's inlet region, the flow pattern changed. There is a vortex flow near the probe's lower end as shown in Figure 1. The lower gas velocity was found at the end of the heat transfer probe, which ranges from 0.057 m/sec to 0.023 m/sec.

The radial air velocities changed frequently because of the bed voidage and gas pressure drop as shown in Figure 1. It is believed that the increase of superficial gas velocity resulted in the increasing local velocity at the radial direction of the CFB riser [2].

1.2 The Effect of Air Pressure

Figure 2 shows the static pressure profiles at k=23 of the vertical direction in the slide plate. The positive numbers represent the tube pressure and it is larger than that of the system pressure.

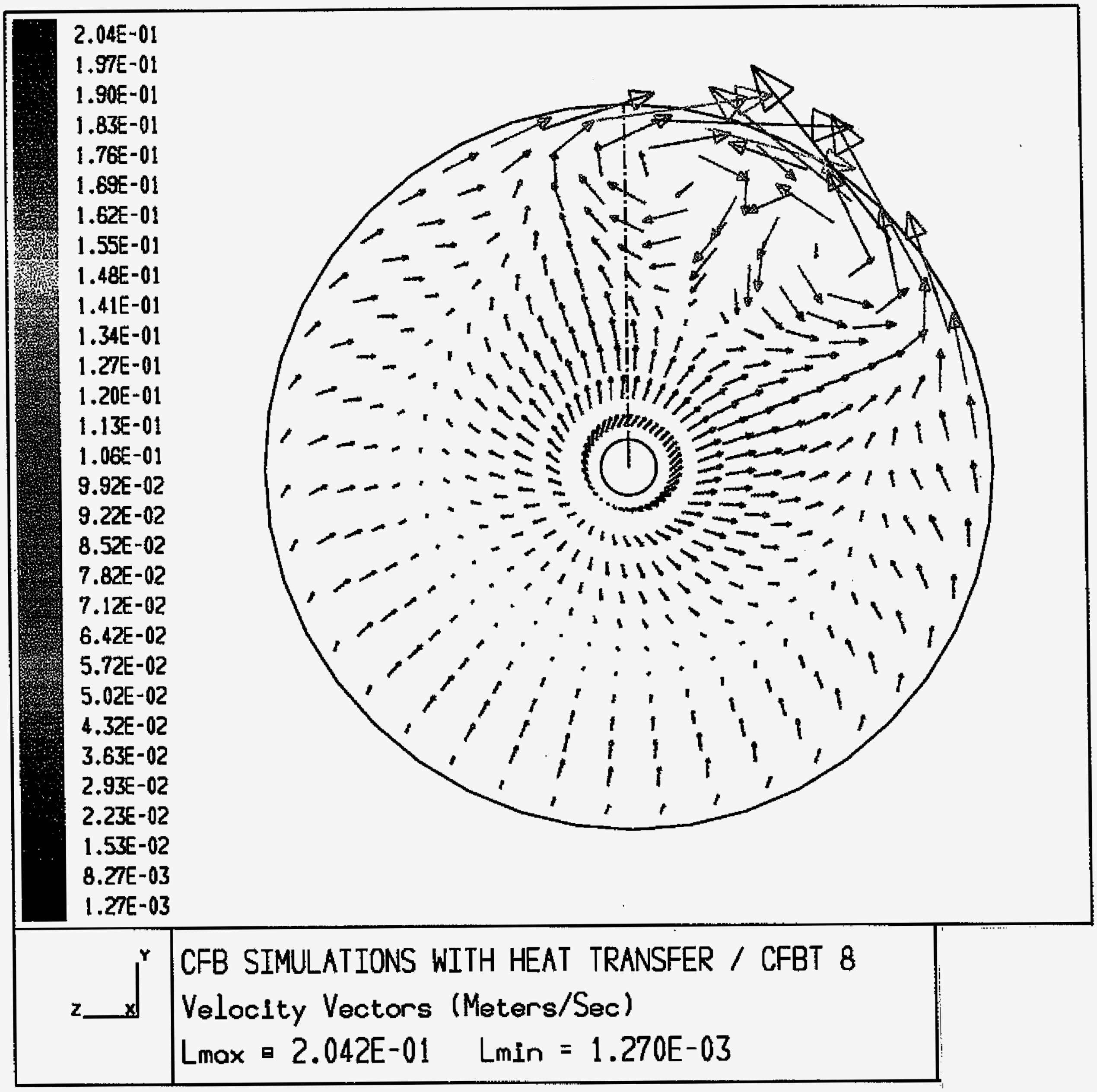

Figure 1 Velocity Profiles at Level, k=23 (Heat Transfer Probe Inlet Location)

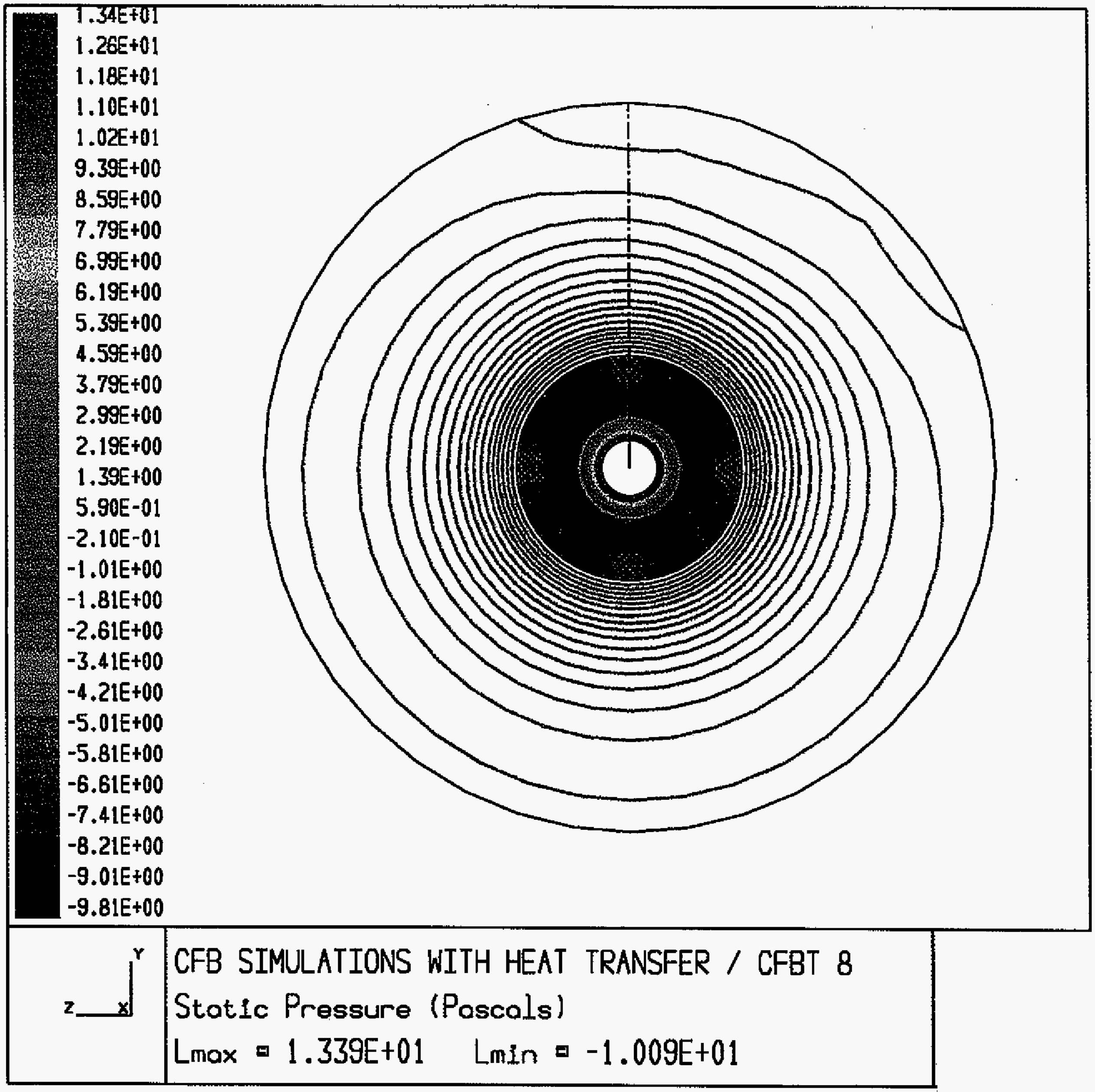

Figure 2 Static Pressure Profiles at Level, k=23

We recognize that a lower pressure zone was formed near the chamber's center region. According to the Bernoulli equation [3], the increasing velocity was coming from pressure potential energy, which was transferred into kinetic energy [4]. It is worth nothing that a higher pressure zone was formed near the surrounding of the secondary air flow. Thus a dead zone or a local swirling flow was formed near the wall region.

1.3 The Effect of Temperature

Figure 3 shows the temperature profiles at k=23 of the vertical direction in the slide plate. We assumed with the heat absorbing water-cooled heat transfer probe enclosing the CFB riser of the cold model. The lower temperatures were found near the CFB riser wall side, which indicated a similar temperature profile along the vertical direction in the CFB chamber [2].

In Figure 3, the temperature was changed from 353 K to 317 K along the radial direction of the CFB chamber, this was the heat transfer inlet level. In the radial direction, the gas temperature near the center region was higher than that near the riser wall zone because of the dilution effect of secondary air injection and the water-cooling effect of the heat transfer probe. The temperature difference between the riser center and the wall was about 25 K to 36 K.

1.4 Heat Transfer Characteristics

Figure 4 shows the heat flux profiles at k=23 of the vertical direction in the slide plate. During this calculation, it was assumed that the heat transfer probe when in contact with gas flow was water-cooled. The heat flux was changed along the radial direction of the CFB riser as shown in Figure 4. The higher heat flux zone was formed near the chamber's center region. It is believed that the influence of bed temperature on local heat transfer is the most important factor.

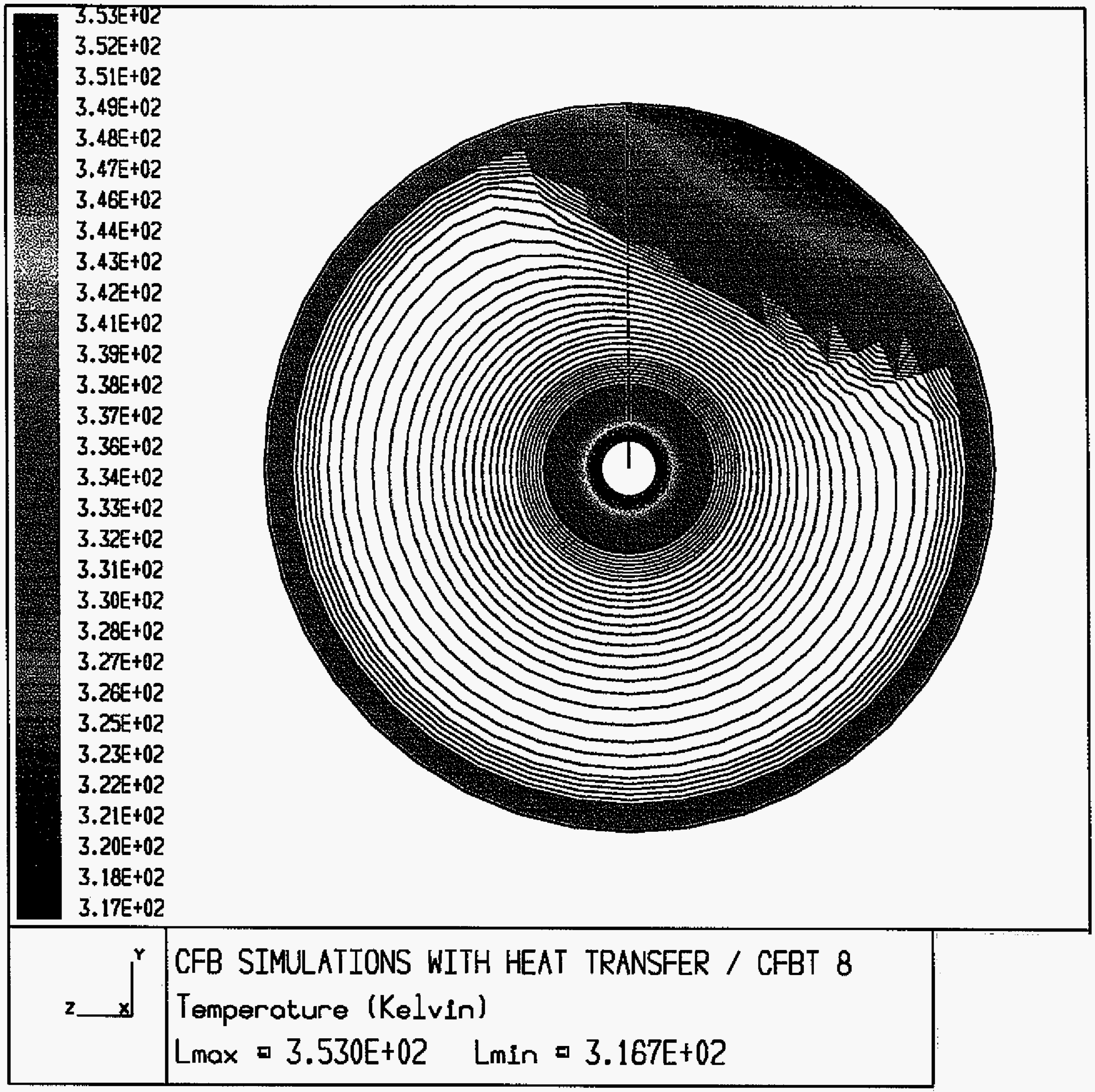

Figure 3 Temperature Profiles at Level, k=23

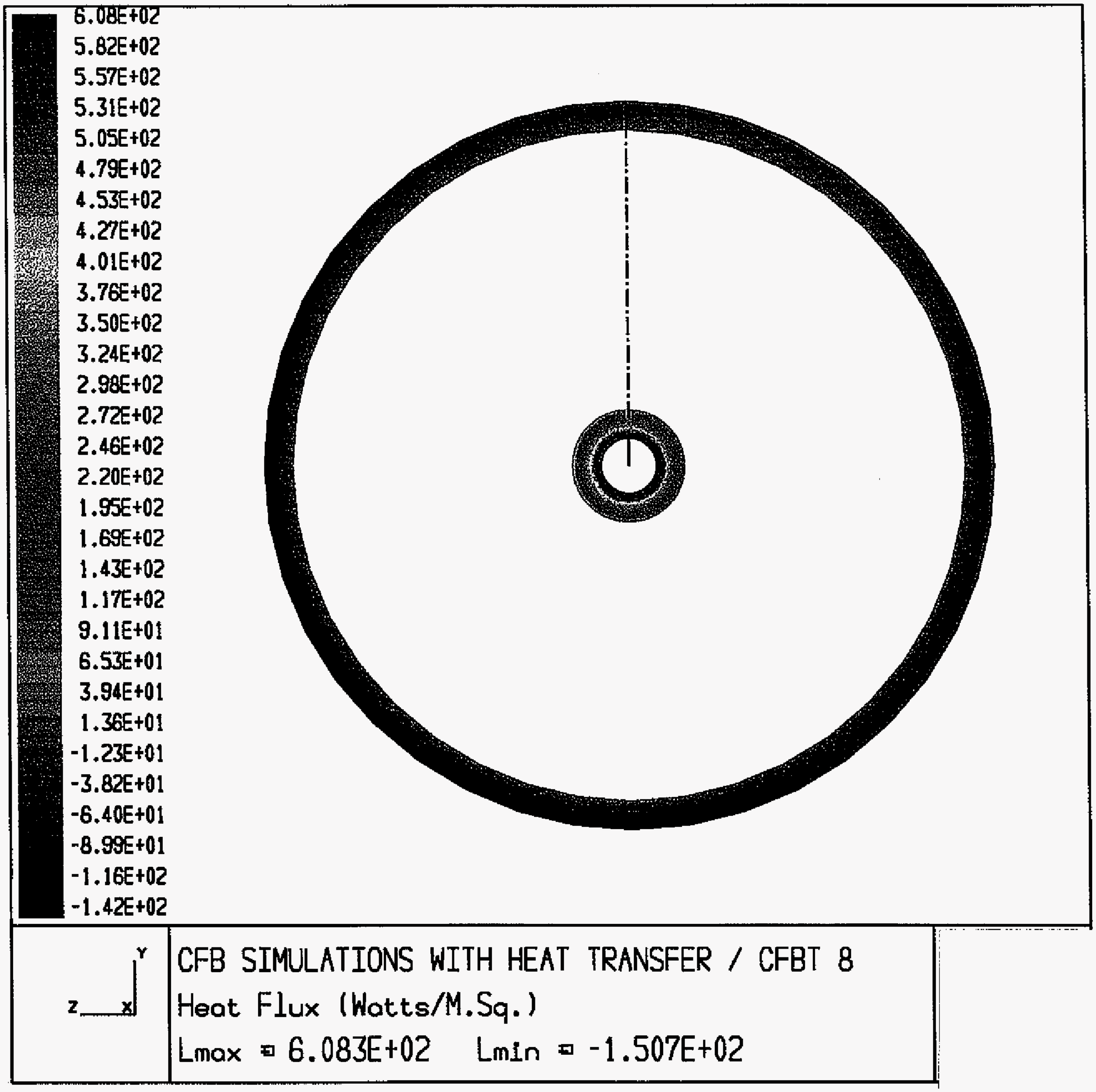

Figure 4 Heat Flux Profiles at Level, k=23

SECTION 2

DESIGN AND ARRANGEMENT OF THE HEAT TRANSFER PROBE

The main column of a circulating fluidized bed normally consists of a core of uniformly dispersed solids which move up with the upper flowing gas surrounded by a thin annuls where the solid clusters usually move down [5]. Thus the mechanism of bed to wall heat transfer in these two regions need to be studied. In the annuls region, the wall of the heat transfer surface usually contacts with the clusters and dispersed phase. The heat transfer coefficient in the core region might include the heat transfer coefficient due to the dispersed phase and the gas phase.

As shown in Figure 5, the bench-scale CFB model and supporting auxiliary subsystems are including the air supply system, instrumented riser, gas-solid separation system, and solid return/storage tank. The air supply subsystem consists of blower, control valves, PVC piping, and flow measuring devices.

A 13 mm O.D. and 343 mm long copper tube will be used as the heat transfer tube to measure the heat transfer coefficients at different axial and radial locations. In order to calculate the heat transfer rate, the heat transfer probe will be placed near the wall. The probe will be placed 57.2 cm above the distributor plate as shown in Figure 5. Hot water will be passed through the tube and the temperature drop of water will be noted at different locations using thermocouple and thermometer. The heat transfer rate will be measured without the bed particles at first stage.

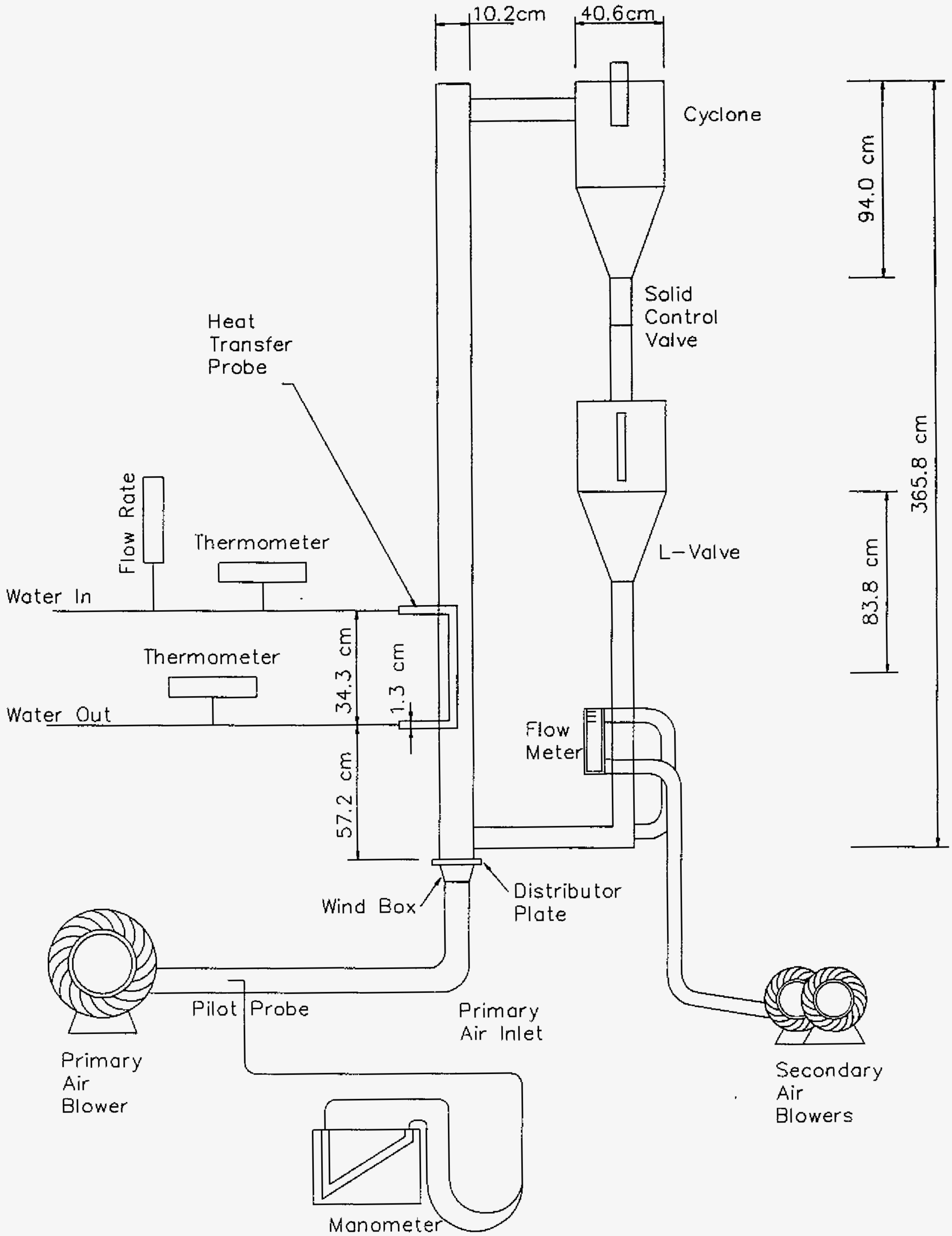

Figure 5 Schematic Diagram of the Bench-scale CFB system with the Heat Transfer Probe

SECTION 3

CONCLUSIONS

Numerical simulation was used to predict the air velocity and pressure, temperature, and heat transfer characteristics in the CFB system. The flow pattern changed when the air flow reached the heat transfer probe's inlet region at level, k=23. In addition, the radial air velocities changed frequently because of the voidage and gas pressure drop at the same region.

The lower pressure zone was formed near the chamber's center zone. The higher pressure zone was formed at the surrounding of the secondary air flow.

The temperature profiles showed the temperature changes near the CFB riser wall side and the center region. the temperature difference between the riser center and the wall was about 25 K and 36 K. The heat flux was changed along the radial direction of the CFB riser. The higher heat flux zone was formed near the chamber's center zone.

The heat transfer probe was designed and installed to measure the heat transfer coefficients at the different axial and radial locations in the CFB riser. A model will be proposed to predict the heat transfer coefficients along the axial and radial variation of the bed in the CFB system.